IMPERIAL COLLEGE OF SCIENCE, TECHNOLOGY AND MEDICINE
Exhibition Road, London SW7 2BX.

ABSTRACTS of PRESENTATIONS

at

WORKSHOP

on

UNSTEADY and TWO-PHASE-FLOWS

Sponsored by U.S. Army Research,
Development & Standardization Group (UK)

28, 29 June 1990

U.S. Army Workshop at Imperial College:

IMPERIAL COLLEGE
Mechanical Engineering - Room 703

Introductory Remarks

Thursday, 28 June 1990

9.00	Introductory Remarks	

Unsteady and two-phase flows

	R. Reichenbach	*Chairman*
9.15	A. Brown and K. Knowles	Unsteady fluid dynamics
9.45	M. Sommerfeld	Shock wave propagation in a dusty gas.
10.15	*Coffee break*	
10.45	A.D. Gosman	A calculation method for unsteady, two-phase flows.
11.15	A.M.P.K. Taylor	Methods for measuring two-phase flow properties
11.45	H. McDonald	Calculation of two-phase flows.
12.15	M. Oldfield	Unsteady pressure and heat-transfer measurements.
13.00	***Lunch***	

Gun flows

	A. Brown	*Chairman*
14.30	G. Keller	Interior ballistic effects of kinetics, grain fracture and viscosity.
15.00	R. Heiser	Interior ballistic flow modelling at the Ernst Mach Institute.
15.30	*Coffee break*	
16.00	G. Klingenberg	Unsteady flow and combustion phenomena.
16.30	R. Perez-Ortiz	Velocity, pressure and heat transfer in a gas gun
17.00	B. Lawton	Instantaneous heat transfer in gun barrels.
19.00	***Dinner***	

X

form 50 per

Friday, 29 June 1990

Unsteady and two-phase flows

	G. Bergeles	*Chairman*
9.00	K.C. Schadow and E. Gutmark	Passive and active combustion control.
9.30	M.N.R. Nina	Passive control of combustion oscillations.
10.00	L. De Luca	Burning stability of solid propellants.
10.30	*Coffee break*	
11.00	F.E.C. Culick	Pressure oscillations in liquid-fuelled propulsion systems.
11.30	E.J. Mularz	Report oOn status of hydrogen fuel-shear-layer experiment.
12.00	J.A.G. Aston and A.D. Crowley	On the flow structure in two-phase shock tubes.
12.30	T. Cebeci	Calculation of unsteady flows with separation.
13.00	***Lunch***	

Unsteady and two-phase flows

14.30	C. Zoltani	Investigation of confined, two-phase, subsonic jet flow.
15.00	L. Carr	Experiments with oscillating airfoils.

Gun flows

	R. Ogorkiewicz	*Chairman*
15.30	F. Seiler and K. Zimmerman	A study on gun erosion in a 20mm-caliber gun tube.
16.00	M. Mach	Studies on secondary combustion of gun exhaust plumes.
16.30	*Coffee Break and Discussion.*	
17.30	Closure	

[redacted]

U.S. Army Workshop at Imperial College, 28-29 June 1990

Unsteady Fluid Dynamics Research at RMCS Shrivenham

A Brown[a], M V Finnis[b], D Bray[c], K Knowles[c], G M Moss[d] and S B Ellis[e]

School of Mechanical, Materials and Civil Engineering
Royal Military College of Science, Shrivenham, Swindon, SN6 8LA

Abstract

Unsteady fluid dynamics and aerodynamics is an area of major interest at RMCS. Within this general area research covers boundary layer transition on flat and curved surfaces in the presence of pressure gradients, jet aerodynamics, vortex aerodynamics and ballistics. All of this work is relevant to the performance of military equipment, particularly gas turbines, helicopters and jet-lift aircraft, guided weapons and tube-launched projectiles.

In laminar boundary layers it is generally accepted that under zero-pressure gradient conditions instabilities can occur at a critical point where the Reynolds number based on momentum thickness has a value of about 160. As this Reynolds number reaches 200, turbulent spots may occur which grow and lead to a fully turbulent boundary layer. Freestream turbulence plays a part in the onset and extent of transition, as does surface vibration. Recent work has investigated the relationship between the frequency and amplitude of surface vibrations and the onset of transition[1]. Of particular interest in turbomachinery applications, however, is the influence of pressure gradients on boundary layer transition. Extensive experimental and theoretical work has been carried out for both adverse and favourable pressure gradients. The work was aimed at a better understanding of heat transfer and skin friction on turbine blade surfaces but is equally applicable to flow over any curved surface.

In the case of convex curved surfaces, the most successful theoretical approach has been found to be a modification of the STAN5 program from Crawford and Kays[2]. An important characteristic of any boundary layer prediction program is the transition function: the original version of STAN5 contained a transition function of a sinusoidal nature. This could be called into action at a pre-specified critical momentum-thickness Reynolds number, usually at a value of 200. At this point the turbulent stresses were gradually increased from zero to their fully-turbulent value at the point where the momentum-thickness Reynolds number was equal to twice the critical value. Roberts and Brown3 developed an alternative transition function based on a velocity gradient factor[4,5] and the total strain rate up to the position of start of transition. This fitted well with their measurements on the suction surface of a gas turbine blade in a cascade. The foregoing, when coupled with a function of freestream turbulence intensity, helped in predicting heat transfer rates between the hot gases and the blade surface.

---

a Professor, Chairman of School and Head of Thermal Power Group.
b Research Officer, Thermal Power Group.
c Lecturer, Aeromechanical Systems Group.
d Senior Lecturer, Aeromechanical Systems Group.
e Senior Research Officer, Aeromechanical Systems Group.

On concave surfaces the boundary layer is subjected to an instability due to centrifugal forces. This results in a secondary flow in the form of streamwise counter-rotating vortices. These Goertler vortices are known to be a modulator of transition to turbulence and the boundary layer prediction programs currently available have a poor performance in such flows. Experimental and theoretical work is currently in hand to study this flow pattern[6,7]. Predictions of streamwise velocity profiles and ranges of vortex amplitude agree well with experimental measurements for zero-pressure gradient. Predictions for Falkner-Skan flows indicate an increased stability to Goertler vortices for favourable pressure gradients and decreased stability for adverse gradients.

Jet aerodynamics research at RMCS is currently concerned with two topics: propulsion nozzle mass flow control using swirl and impinging jets in cross-flow. The former has involved an analytical study[8] which has demonstrated the feasibility of the scheme[9]; this is not, however, an unsteady flow phenomenon. Impinging jets, on the other hand, are a well-known source of flow unsteadiness[10].When a jet impinges on a surface normal to its axis a wall jet is formed which spreads out radially from the impingement point. If a cross-flow (parallel to the wall) exists then the wall jet will eventually stagnate and roll up to form a vortex, arranged as a horse-shoe about the jet (see Fig 1). This flow-field is of particular interest in the development of V/STOL aircraft, where the cross-flow may be due to ambient wind or aircraft motion. With multiple jets there is the added complexity of the fountain formed by the two opposing wall jets.

The mean position and strength of the ground vortex are important due to their effects on aircraft pressure loads and air intake flows (where there is the specific problem of hot gas ingestion). Some of the factors which affect the ground vortex position have been investigated previously but there is considerable scatter in the experimental data and very little high jet Mach number data. The current project[11] has been looking at the influence of nozzle height (2 to 8 diameters), nozzle pressure ratio (1.05 to 4), cross-flow-to-jet velocity ratio and vector angle (+10 to -20 degs) for single- and twin-jet configurations with both fixed and moving ground planes (simulating ambient wind or aircraft motion). For the twin-jet configurations the effect of nozzle splay is also being investigated. The programme of work is both experimental and numerical, a particular aim being to determine the difference in ground vortex position with a moving, rather than fixed ground plane. This information is needed before wind tunnel tests are conducted on specific V/STOL aircraft configurations in ground effect.

An extensive series of experiments is being conducted in the RMCS low-speed, open-jet wind tunnel, which is equipped with a rolling-road (1.6m x 1m) and a compressed air supply (to single or twin 25mm diameter nozzles). Tests have consisted of pitot-static probe traverses a few millimetres above the ground plane, upstream of the ground vortex in the plane of symmetry. This has allowed the vortex penetration to be quantified (see Fig.2) and plotted against parameters such as cross-flow-to-jet velocity ratio (Fig.3). Comparisons with fixed ground board statics showed this approach to be perfectly satisfactory (except directly under the vortex core where the probe, not surprisingly, gave a lower static pressure). A correlation has been derived for the positions of ground vortex core, separation point and maximum penetration point[12] (see Fig.2).

Flow visualisation has been conducted using smoke and laser light-sheet. Results for a twin-nozzle arrangement have shown the existence of a "spike" in the plane of symmetry. The ground vortex formed with twin nozzles is seen to be even more unsteady than that due to a single nozzle and the flow-field seems to be very sensitive to cross-flow yaw. Computational work to date[13] has used the PHOENICS finite-volume code to perform 2-D calculations with the standard k-e turbulence

model. Initial results are encouraging (see Fig. 4) and work is proceeding to extend the computations to the full 3-D case.

Vortex aerodynamics has been a subject of research at RMCS for many years due particularly to its importance in guided weapon flight and in helicopter rotor performance. Current work is investigating rotor/fuselage, blade/vortex (BVI) and rotor/tail-rotor interactions. A transverse vortex generator has been built. This consists of a symmetrical aerofoil section (NACA 0018) spanning the 8ft, low-speed test section of the RMCS closed wind-tunnel and capable of being pitched rapidly to a modest angle of incidence (approx. 10°). This sheds a starting vortex which then moves downstream over a test body. The test body, which could be an aerofoil for BVI experiments[14] or a body of revolution for rotor/fuselage interaction experiments, is instrumented with high-frequency pressure switches allowing pressure vs. time maps to be drawn for the pressure-tapped area. The interactions being modelled in these experiments are idealised events, not of themselves representative of real interaction geometries, but capable of providing data for computer code validation or empirical input.

Within the area of aero-ballistics RMCS is investigating the flight of tumbling projectiles as part of a major study which is intended to put onto a proper scientific basis the estimation of safety traces for firing ranges. The areas which are currently being examined include the following.

i) The ricochet of typical projectile shapes from different media, including wet and dry sand, concrete, pebbles, turf and water. There are several aspects to these investigations which are designed to highlight the physical mechanisms involved in the ricochet process and to provide post-ricochet data for subsequent trajectory prediction.
ii) The generation of a full six-degree-of-freedom trajectory simulation for a complete range of attitudes and with no small-angle or linearising assumptions.
iii) The production of aerodynamic data for the above including the effects of round asymmetry caused by impact and aerodynamic parameters for rounds with large angular velocities.
iv) The trim states for asymmetric rounds having varying degrees of axial spin.

The attached figures give some specimen results. Figure 5 is the predicted drag coefficient for an M80 round at a flight Mach number of 2.5, whilst Figure 6 gives some measured speed vs time data for 7.62mm tracer rounds after ricochet. The rate of decay of these is proportional to the effective drag coefficient of the round. The inference is that, in some cases, a tumbling round re-stabilises into low-yaw angle flight.

Figures 7 and 8 show the predicted flight paths for a bent round flying in a dense medium. The changes in the sign of the heading angle are particularly noteworthy - if this were considered to be the flight path in the sand of a stop butt, the round could subsequently emerge flying either to the right or to the left of the original azimuth heading.

References

1. Knowles, K and Davis, JA, "An experimental study to determine the cause of transition on a flat plate", EUROMECH 228 Colloquium on Boundary Layer Instability and Transition, Exeter, Sept 1987.

2. Crawford, ME and Kays, WM, "STAN5 - A program for numerical computation of two-dimensional internal and external boundary layer flows", NASA CR-2742, 1976.

3. Roberts, G and Brown, A, "Boundary-layer transition regions on turbine blade suction surfaces", ASME paper 84-GT-284, 1984.

4. Launder, BE, "Laminarization of the turbulent boundary layer in a severe acceleration", J App Mechs, 31, 707-8, 1964.

5. Jones, WP and Launder, BE, "Some properties of sink flow turbulent boundary layers", JFM, 56, 337-351, 1972.

6. Martin, BW and Brown, A, "Factors influencing heat transfer to the pressure surfaces of gas turbine blades", Int. J. Turbo. and Jet Engines, 1, 247- 258, 1984.

7. Finnis, MV and Brown, A, "Stability of a laminar boundary layer flowing along a concave surface", Trans. ASME, J.Turbomachinery, 111, 376-386, Oct 1989.

8. Knowles, K and Carpenter PW, "Subcritical swirling flows in convergent, annular nozzles", AIAA Jnl., 27, 2, 184-191, Feb 1989.

9. Knowles, K and Carpenter PW, "The use of swirl for flow control in propulsion nozzles", AIAA paper no. 88-3003, 1988.

10. Rockwell, D and Naudascher, E, "Self-sustained oscillations of impinging free shear layers", Ann. Rev. Fluid Mech., 11, 67-94, 1979.

11. Bray, D and Knowles, K, "Normal impinging jet in cross-flow - a parametric study", AIAA paper no. 89-2957, 1989.

12. Ing, D and Bailey, P, "Model RA117 - data analysis of Phase 2 jet in cross -flow and ground effect tests in the Warton 5.5m low speed wind tunnel", Logico Systems rept. no. 18, June 1989.

13. Knowles, K and Bray, D, "High Mach number impinging jets in cross-flow - a comparison of computation with experiment", in: "Numerical Methods in Laminar and Turbulent Flow", 6, ed. Taylor, C et al, 1389-1398. Pub Pineridge Press, 1989.

14. Straus, J, Renzoni, P and Mayle, RE, "Airfoil pressure measurements during a blade-vortex interaction and a comparison with theory", AIAA Jnl., 28, 2, 222-8, Feb 1990.

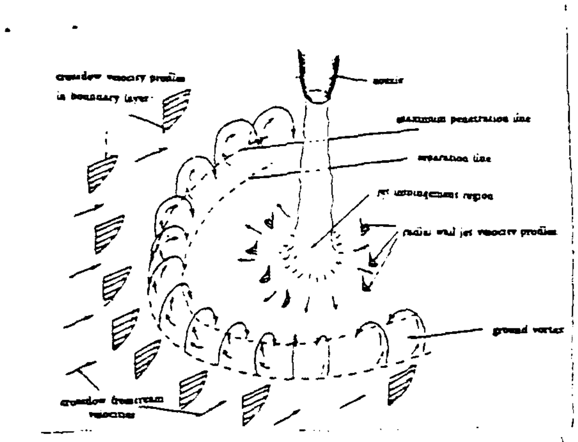


Figure 1 Impinging Jet in Crossflow—Stationary Groundplane

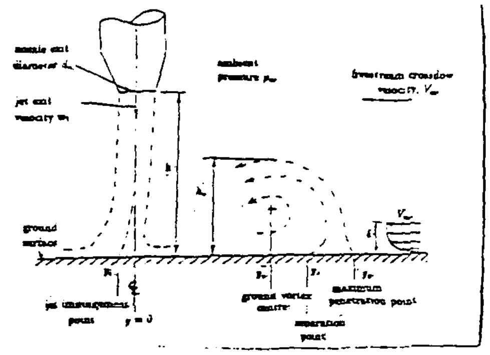


Figure 2a Horseshoe Ground Vortex formed by Impinging Jet in Crossflow — Two-dimensional view

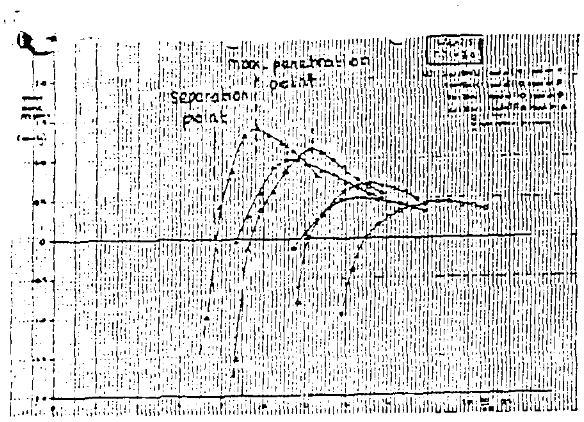


Figure 2b Derivation of Penetration Distances from typical Static Pressure Plots

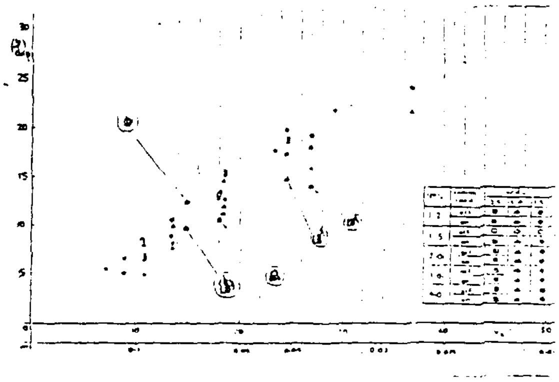


Figure 3 Penetration Distance Plotted Against Effective Velocity Ratio

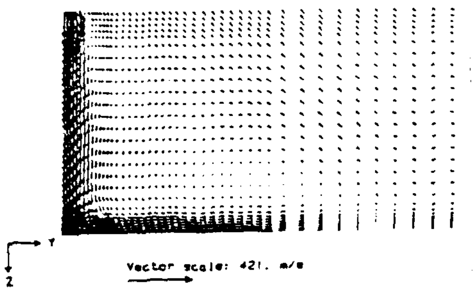


Figure 4a Typical PHOENICS Velocity Vector Plot of Jet in Crossflow and Ground Effect

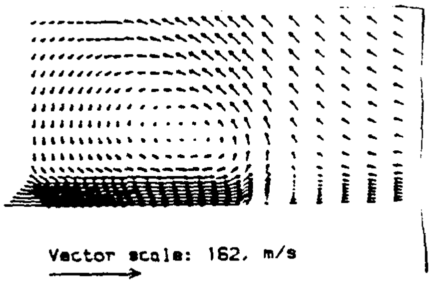


Figure 4b Enlarged Plot of Ground Vortex Region

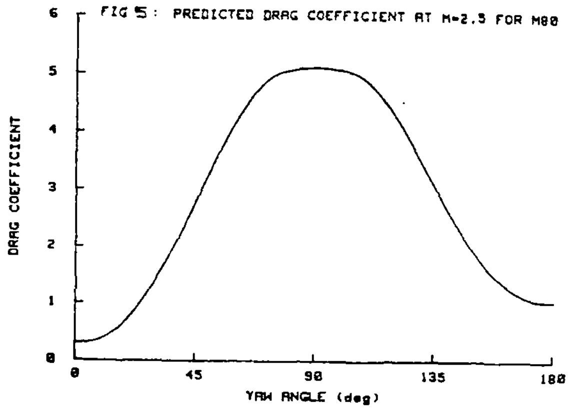


Fig 6 : measured speed decay after ricochet

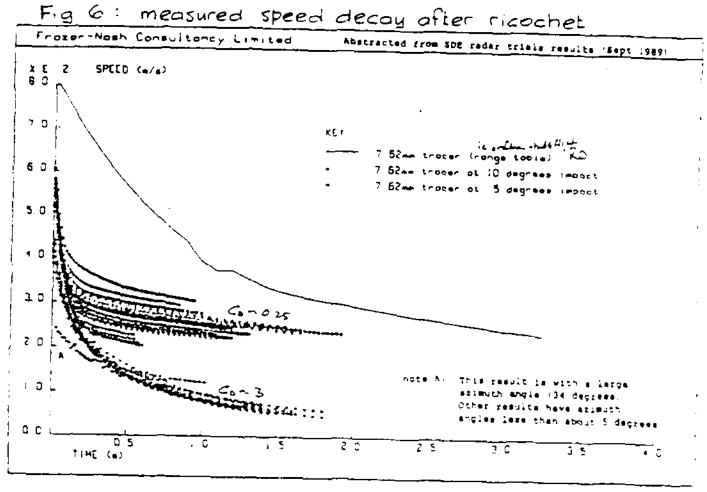
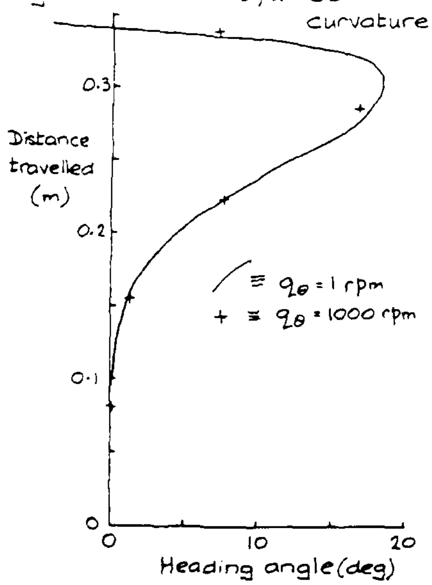
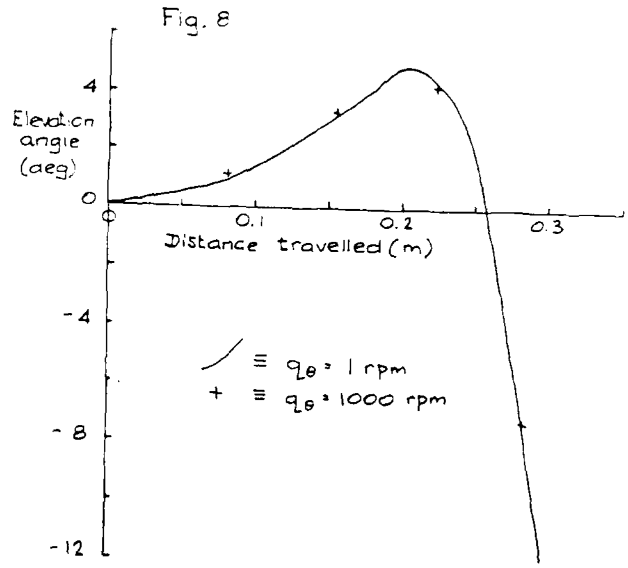


Fig 7: Bent round, 20° centreline curvature





SHOCK WAVE PROPAGATION IN A DUSTY GAS

Martin Sommerfeld
LSTM-Erlangen
Universität Erlangen-Nürnberg
Egerlandstr. 13, D-8520 Erlangen

The shock propagation in a gas-particle mixture exhibit certain characteristics due to the relaxation phenomena associated with the dispersed particles. This results in a relaxation region behind the shock wave and a decay of the shock wave propagation into a dusty gas. The decay rate is dependent on the shock strength and the properties of the particles e. g. size, material density and particle loading. Experimental results showed that the shock strength initially is rapidly decaying and then approaching an equilibrium value. This equilibrium value is only dependent on the shock strength and the particle loading.

Experiments and one-dimensional numerical simulations will be presented which show the above described phenomena. Furthermore, results will be discussed which consider two-dimensional effects of shock wave propagation in dusty gases, e. g. boundary layers shock wave reflection at wedges and area constrictions.

A Calculation Method for Unsteady Two-Phase Flows

A D Gosman
Mechanical Engineering Department
Imperial College of Science Technology & Medicine
London SW7 2BX

ABSTRACT

An outline is provided of CFD methodology for predicting unsteady, disperse, two-phase flows with accompanying heat and mass transfer; and sample applications are presented. The method is based on a continuum description of the continuous phase and a Lagrangian model of the discrete phase, which may consist of particles, droplets or bubbles. Models are included to represent turbulence effects on both phases and the processes of breakup, collision and coalescence of the discrete phase.

The governing equations are solved by a fully-implicit algorithm based on the existing PISO method for single-phase flows. The two-phase version contains provision for the strong mass, momentum and thermal energy interchanges which may take place between the phases, and also incorporates a temporal subcycling procedure to improve accuracy and efficiency in circumstances where the discrete phase velocities are appreciably higher than those of the continuous phase.

The performance of the methodology will be demonstrated through extracts from validation studies for pulsed liquid sprays produced by a pressure atomiser, as used in reciprocating engines.

U.S. ARMY WORKSHOP AT IMPERIAL COLLEGE

Thursday 28 June 1990

UNSTEADY & TWO-PHASE FLOWS

"Optical Methods for measuring two phase flow properties"

A. M. Taylor, Department of Mechanical Engineering, Imperial College

Abstract

The presentation will provide an overview of the optical methods which exist for the measurement of particle size, velocity, size-velocity cross-correlation and particle flux and concentration in dispersed two-phase flows. The available instruments can be conveniently categorized into two types: 'integral techniques' and 'single particle counters'. A well-known example of the integral technique is the Fraunhofer-diffraction instrument for the measurement of particle size distribution integrated along the direction of propagation of a laser beam. The technique can size in the range between about 1 and several hundred μm at maximum concentrations of the dispersed phase limited to obscuration of about 50% without corrections being required. The single particle counters (SPCs) make measurements of the size of individual particles in a test volume which is generally smaller than 1 mm^3. SPCs are frequently combined with laser-Doppler velocimeters to provide simultaneous measurements of size and velocity and hence cross correlation also. SPCs can be further sub-divided into methods for non-spherical and spherical particles. Non-spherical particles can be sized using either the amplitude or the visibility of the Doppper signal in the range between about 1 - several hundered μm and about 5 - 100 μm respectively. The accuracy of measurement depends on both the concentration and shape of the particles. Spherical particles can be conveniently sized by a third variation of the SPC, namely a phase-Doppler anemometer. The basis of this technique is that the droplet acts as a 'magnifying glass' for the fringes in the test volume of the anemometer and the instrument measures the magnification and hence infers the droplet diameter. Sizes in the range from a few to several hundered μm can be measured and at concentrations which have resulted in obscuration of the beams in excess of 30%. Some examples of measurements will be provided.

CALCULATION OF THE TWO-PHASE FLOWS USING AN EULERIAN-LAGRANGIAN ANALYSIS

Jayant S. Sabnis, Frederik J. de Jong
and Henry McDonald

Scientific Research Associates, Inc.
50 Nye Road P.O. Box 1058
Glastonbury, CT 06033

Computational techniques used in simulation of two-phase flows can be broadly categorized into two approaches, viz. the Eulerian-Eulerian analysis and the Eulerian-Lagrangian analysis. Both techniques involve computing the continuous phase using an Eulerian analysis. The influence of the particulate phase (either particles or droplets) on the continuous phase is accounted for by inclusion of inter-phase coupling terms in the Eulerian equations, which in the absence of these terms would be the usual Navier-Stokes equations. The particulate phase, on the other hand, may be treated with either a continuum or a discrete model. The Eulerian-Eulerian technique uses a continuum model for the particulate phase and is commonly termed the two-fluid model. This approach models a dense granular bed very conveniently and this undoubtedly accounts for its popularity in modeling gun interior ballistics where large particle loading ratios occur over most of the cycle (see for example Gough[1] and Gibeling et al.[2]). The second approach employs Lagrangian description to analyze the motion of the particulate phase using computational "particles" to represent a collection of physical particles (or droplets). Newton's law of motion is employed to simulate the particle motion under the influence of the local environment produced by the continuous phase.

In simulation of evaporating sprays, it becomes necessary to account for the fact that the discrete is phase not mono-dispersed. To accomplish this in the Eulerian-Eulerian methodology, the two-fluid model can be generalized into a multi-fluid model. However, the CPU time requirements increase rapidly with the increasing number of particle size classes since an extra "fluid" has to be added for every particle size class. Also, numerical dissipation and dispersion could be a major problem where it might be necessary to maintain sharp inter-phase boundaries. The Eulerian-Lagrangian analysis treats the particl size as one of the attributes assigned to computational particles and hence has no trouble simulating flows which involve changing particle size. Further, this approach involves integration of ODE's

for the particulate phase and hence is numerically efficient, in addition to eliminating the numerical dispersion. The deterministic nature of the particle dynamics allows incorporation of models for turbulent dispersion, agglomeration, collision, etc.

In Eulerian-Lagrangian algorithms, the coupling terms for the continuous phase equations can be computed from the knowledge of the trajectories for the representative particles and their attributes at the intersection of the trajectories with Eulerian cell boundaries. The coupling terms can also be computed from the instantaneous distribution of the computational particles in the domain. The former approach has been employed, for example, by Crowe, et al. [3] and Gosman and Ioannides [4], while the later approach has been used by Duckowicz [5] and Sabnis, et al. [6]. The particle trajectory models are suitable for simulation of steady flow problems only, while the particle distribution models can be used for steady as well as unsteady flow simulations. The algorithm developed by Sabnis, et al. [6] has employed transformation of coordinates to integrate the equation of motion of the particle in Eulerian computational space, and coupled it with a linearized block implicit scheme of Briley and McDonald [7], resulting in an efficient computer code for two-phase flow simulation including sub-layer resolution capability without the need to use wall-functions.

The present paper describes the application of the CELMINT (Combined Eulerian-Lagrangian Multidimensional Implicit Navier-Stokes Time-Dependent) code of Sabnis, et al. [6] to simulate two-phase flows with and without evaporation and comparison of the computed results with experimental data. The test cases used in data comparison were the two-phase mixing layer experiment of Milojevic, et al. [8] which involved no evaporation, and the evaporating spray experiment of Solomon, et al. [9].

REFERENCES

1. Gough, P.S.: Numerical Analysis of a Two-Phase Flow with Explicit Internal Boundaries, IHCR 77-5, Naval Ordnance Station, Indian Head, MD, April 1977.

2. Gibeling, H.J., McDonald, H. and Banks, N.E.: An Implicit Numerical Analysis for Two-Dimensional, Two-Phase Turbulent Interior Ballistic Flows. AIAA Paper 84-0561, 1983.

3. Crowe, C.T., Sharma, M.P. and Stock, D.E.: The Particle-Source-in-Cell Method for Gas Droplet Flow. ASME Journal of Fluids Engineering, Vol. 99, No. 2, 1977, pp. 325-335.

4. Gosman, A.D. and Ioannides, E.: Aspects of Computer Simulation of Liquid-Fueled Combustors. Journal of Energy, Vol. 7, No. 6, 1983, pp. 482-490.

5. Dukowicz, J.K.: A Particle Model for Liquid Sprays. Journal of Computational Physics, Vol. 35, No. 2, April 1980, pp. 229-253.

6. Sabnis, J.S., Gibeling, H.J. and McDonald, H.: A Combined Eulerian-Lagrangian Analysis for Computation of Two-Phase Flows. AIAA Paper 87-1419, 1983.

7. Briley, W.R. and McDonald, H.: Solution of Multidimensional Compressible Navier-Stokes Equations by a Generalized Implicit Method. Journal of Computational Physics, Vol. 24, No. 4, 1977, pp. 372-397.

8. Milojevic, D., Borner, Th. and Durst, F.: Prediction of Turbulent Gas-Particle Flows Measured in Plane Confined Jet. Proceedings of the World Congress on Particle Technology, Nurnberg, April 1986, pp. 485-505.

9. Solomon, A.S.P., Shuen, J.-S., Zhang, Q.-F. and Faeth, G.M.: A Theoretical and Experimental Study of Turbulent Evaporating Sprays, NASA Contractor Report 174760, September 1984.

10. Schlichting, H.: Boundary-Layer Theory, Seventh Edition, McGraw Hill Book Company, New York, 1979, pp. 729-750.

THE INTERIOR BALLISTIC FLOW MODELLING

AT THE ERNST-MACH-INSTITUTE

R. Heiser

Fraunhofer-Institut für Kurzzeitdynamik
- Ernst-Mach-Institut -
Hauptstrasse 18, D-7858 Weil am Rhein

Abstract

The activities in interior ballistics modelling at the Ernst-Mach-Institut are presented. Two different types of flows are addressed:

1. The AMI code covers the multidimensional two-phase flow of a granular charge. Arbitrary geometries of the flow region as well as igniting devices on the boundaries are included now. Eulerian equations are used.

2. The multidimensional turbulent one-phase flow is investigated by an extension of the DELTA code, an Navier-Stokes solver.

For both cases, the theoretical basis as well as some numerical results are presented.

Fraunhofer-Institut für Kurzzeitdynamik
Ernst-Mach-Institut
Abteilung für Ballistik

INVESTIGATION OF UNSTEADY FLOW AND COMBUSTION PHENOMENA

G. Klingenberg

ABSTRACT

Prepressurized mixtures either of hydrogen/oxygen or hydrogen/carbon monoxide/oxygen, with helium as diluent, are filled into a windowed combustion chamber of a single-stage gas gun of caliber 20 mm and electrically ignited. The interior ballistics flow and combustion phenomena are studied by means of pressure, temperature, concentration, and gas velocity measurements. Infrared emission spectroscopy is employed to determine local gas temperature and concentration profiles of chemical species. A multi-channel spectrometer is used to measure simultaneously the IR radiation of the chemical species. The results presented provide input data for unsteady fluid dynamics models.

U.S. Army Workshop at Imperial College

Velocity, pressure and heat transfer in a gas gun

R. Perez-Ortiz

The evolution of gun simulators at Imperial College has led to the development of the Mark III simulator which was designed to increase the projectile and gas velocities to supersonic speed. The Mark III rig simulates a constant-diameter preburnt propellant ideal gas gun. It is designed to operates horizontally and permit initial pressures form 10 to 60 bars.

Gas velocity measurements were obtain by a dual-beam laser-Doppler velocimeter at four locations from the breech and with initial pressure of 10 bar gauge, corresponding to muzzle velocity of 220 m/s. Indication of a turbulent wall boundary layer thickness ranging from 26% to 43% of the bore was found. For initial pressure of 10 to 60 bars corresponding to muzzle velocities of 220 to 420 m/s, the breech pressure was measured by a piezo-electric transducer and the surface temperature with a fast response thin-film resistance thermometer at seven locations along the tube. Comparison of measured projectile position and that calculated by assuming quasi-steady one dimensional isentropic expansion shows good agreement. Measured temperature-time profiles at different locations and the same pressure show that the temperature change due to compression heating increases with distance from the breech and the temperature change at the end of the shot reduces at locations far from the breech. At the same position the wall-temperature change increases for higher pressures. Preliminary heat flux calculations show that the heat flux is high immediately after the projectile passes the measuring position and subsequently decreases to a constant value. The maximum heat flux occurs at locations near the breech.

Passive and Active Combustion Control

K. C. Schadow and E. Gutmark
Research Department
Naval Weapons Center
China Lake, CA 93555-6001

Combustion characteristics of dump combustors, including flammability limits, occurrence of combustion instabilities, and combustion efficiency, are closely related to the shear-flow dynamics associated with flow separation at the downstream-facing step into the combustor. With the detailed understanding of the shear-layer development from initial flow instabilities into large-scale structures and subsequent breakdown into fine-scale turbulence, it is possible to passively and actively control the combustion characteristics. Passive combustion control is achieved by changing the initial shear-flow conditions at the dump using nonstandard inlet duct cross sections. Active combustion control is obtained by acoustic excitation of the initial shear layer and periodic fuel modulation. The control experiments, which resulted in reduced pressure oscillation amplitudes and extended flammability limits, were conducted in three steps of increasing physical complexity. Nonreacting experiments (hot-wire anemometry, flow visualization) were followed by combustion experiments in laboratory flames (Planar Laser Induced Fluorescence (PLIF) imaging technique) and dump combustors. The combustion control experiments with dump-stabilized flames are being extended to disk-stabilized flames. Also, in addition to dump combustors, the critical role of large-scale structures in driving pressure oscillations in a regenerative liquid propellant gun is being investigated.

PASSIVE CONTROL OF COMBUSTION OSCILLATIONS

by

Mário Nery R. Nina

Instituto Superior Técnico

Lisbon

An experimental investigation has been conducted in bluff-body stabilized premixed propane flames in pipes.

Measurements of wall pressure were performed with a Kistler probe located in the cold pipe.

Results obtained with different pipe lenghts and stabilizer geometries are presented, as a function of equivalent ratio and velocity, in order to characterise the domains of rough combustion and the stability limits. The results obtained with the disc were used as reference case.

NONLINEAR INTRINSIC BURNING STABILITY OF SOLID PROPELLANTS

by

Luigi De Luca
Dipartimento di Energetica and CNPM/CNR
Politecnico di Milano
32 Piazza Leonardo da Vinci, 20133 Milano, Italy

ABSTRACT

Intrinsic burning stability of solid propellants is tightly connected with the flame structure of the tested propellant. This implies the knowledge of the fundamental thermokinetics not only in the gas-phase, but also at the burning surface and often in the condensed-phase. In addition, the effects of the energetic sources driving intrinsic instability may be significantly increased or damped by the thermophysical properties of the propagating medium. Again, this implies an accurate enough knowledge of the relevant properties in dependence of local temperature and perhaps pressure.

Stability of composite and double-base solid propellants is discussed within a nonlinear framework (finite size disturbances of thermal profiles due to fluctuations of pressure and/or external radiation). The basic assumptions of 1D and quasi-steady gas-phase are invoked. Specific submodels of overall thermokinetics and proper datasets of thermophysical properties are presented for the condensed-phase and gas-phase of the two families of propellants under study. Double-base compositions are less stable than composite propellants due to a thick fizz zone in the gas-phase, extended reactions in the condensed-phase, and also less heat storage.

Transient burning, computed by numerical methods under a variety of operating conditions, allows to check the analytical stability predictions and to obtain the complete time solutions of the governing set of equations. Time-invariant, self-sustained steady oscillations, and the trivial unreacting state are the possible stationary solutions predicted analytically and verified numerically. Pressure Deflagration Limit and dynamic extinction are also predicted analytically and verified numerically. All of these effects are confirmed experimentally as well. Overstability phenomena, predicted analytically and verified numerically, were experimentally confirmed for double-base propellants subjected to IR laser pulses.

Combustion wave stability is augmented by heat release in the gas-phase and diminished by heat release at the burning surface and/or condensed-phase. Maximum stability is obtained if energy is released in the gas-phase by a diffusion flame anchored to the burning surface.

A Planar Reacting Shear Layer System for Turbulent-Combustion Interactions

by
C.J. Marek, C.T. Chang, B. Ghorashi,
C.C. Wey, C. Wey, E.J. Mularz

NASA Lewis Research Center

Several recent studies and reviews of current research in turbulent reacting flows have concluded a need for additional work in planar free shear layer systems. In the mean time, there is an urgent need for further understanding of the planar mixing layer flow-combustion interaction as well as the need for experimental results to verify the accuracy of existing computational fluid dynamics codes. These experimental data are needed for immediate applications in space and aeronautics projects. For this reason, a very versatile planar reacting shear layer (PRSL) facility for hydrogen and heated air (870K) was built and is in early testing at NASA Lewis Research Center.

The multiple objectives are (1) to establish the effects of initial conditions on jet development above Mach 0.25 for the prediction of the shear layer spreading rate, (2) to identify the effects of heat release on the turbulent flow field of the variable density flows (i.e., density difference between fuel and coflowing air), and (3) to investigate the effect of large scale structures and acoustic excitation interaction on fuel-air mixing in high speed flow.

Multiple measurement techniques may be used to study the flow field simultaneously. Instrumentation includes laser imaging - MIE and PLIF to determine location and species concentration in the reaction zone, schlieren - time resolved density field visualization, LDV - flow field velocity and stress tensor mapping, and acoustics - pressure boundary condition data. A description of the facility will be presented along with the data that has been obtained to date.

On The Flow Structure in Two Phase Shock Tubes

J.A.G.Aston and A.B.Crowley
Computational and Applied Mathematics Group
Royal Military College of Science
Shrivenham
Swindon SN6 8LA

January 29, 1990

Abstract

Numerical methods are employed to investigate the structure of two phase Riemann problems in propellant beds. The model used is one based on the equal pressure model. Since the equations arising from this assumption are known to be ill posed a careful analysis of the solutions is made to determine how this lack of hyperbolicity manifests itself in the results.

The effects of particle size, gas to particle density ratio are also investigated and the results compared to both dusty gas and classical shock tube flows.

Finally the structures calculated are examined for their physical plausibility in describing pressure waves and deflagration to detonation transitions in propellant beds.

PROGRESS IN THE CALCULATION OF UNSTEADY AIRFOIL FLOWS AT LARGE ANGLES OF INCIDENCE

by

Tuncer Cebeci

Aerospace Engineering Department

California State University, Long Beach

Abstract

The presentation will describe the progress made in the extension of the steady interactive boundary-layer method of Cebeci et al.[1] to unsteady flows over practical airfoils subject to a ramp-type motion. The method makes use of the unsteady panel method developed by Platzer and his student, Teng[2], and is able to compute flows with large regions of flow separation. By solving the quasi-steady and unsteady boundary-layer equations in an interactive method, the quasi-steady method will be assessed over a range of angles of attack and frequency in terms of convenience, accuracy, and the computational cost. The calculations will encompass airfoil and wake flows at angles of attack close to the start of the dynamic stall and will provide insight into the development of dynamic stall as a result of the trailing-edge separation.

References

1. Cebeci, T., Clark, R.W., Chang, K.C., Halsey, N.D. and Lee, K.: Airfoils with Separation and Resulting Wakes. J. Fluid Mech., Vol. 173, pp. 323-347, 1986.

2. Teng, N.G.: The Development of a Computer Code for the Numerical Solution of Unsteady Inviscid, and Incompressible Flow Over an Airfoil. M.S. Thesis, Naval Postgraduate School, Monterey, CA, 1987.

A STUDY ON GUN EROSION IN A 20-MM-CALIBER GUN-TUBE

F. Seiler*, K. Zimmermann*, F. Schedlbauer+, V. Schlett#

*Franco-German Research Institute of Saint-Louis (ISL)
F-68301 Saint-Louis, France

+Fraunhofer-Institut für Treib- und Explosivstoffe (ICT)
D-7507 Pfinztal-Berghausen, Fed. Rep. of Germany

#Institute for Applied Material Research (IFAM)
D-2820 Bremen, Fed. Rep. of Germany

ABSTRACT

Since several years at ISL investigations on gun tube erosion have been carried out to become a detailed insight in the action of different propellant charges and distinct modes of additive addition on wear and erosion. During this time a special interest was directed towards the well-known influence of so-called additives on erosion reduction. In this paper some important features of recently done erosion measurements will be presented, showing the erosion behavior of some normal and LOVA propellant charges. The LOVA propellant charges investigated have been receiveded from ICT containing different chemical compositions. The effect of additive on inhibiting erosion was investigated in using titanium dioxide (TiO_2) or wax or a mixture of both with a ratio of 45%/55% (swedish additive). The additive was placed in the combustion chamber on the one hand as a "liner" around the front part of the propelling charge and on the other hand in a "tablet form" just behind the base of the projectile.

For experimental studies of gun erosion single-shot experiments have been carried out in a test-gun-device of 20-mm-caliber. For yielding informations about the erosive action, surface sensors have been used for measuring the surface erosion with so-called Knoop indentations and the erosion of edges or cracks with cut in circular grooves. Surface analytical and metallographical investigations of the surface sensors have been done by IFAM. Besides, the wall temperature was measured using Ni-steel thermocouples.

The erosion measurements have shown that it can be assumed that one parameter influencing the erosive action of a propellant charge is its heat of explosion. It comes out that the erosion increases with rising heat of explosion. The erosion and the inner wall temperatures (depending on the heat of explosion) can be related to each other. This prediction is also valid for the LOVA propellants, which therefore behave with regard to this dependence like the normal propellants.

Towards erosion reduction the best results have been gathered with additive insertion as a liner around the front part of the propellant charge. Moreover, the liner works most effective when a special liner material (TiO_2-support) is used setting free very fast the additive particles during the interior ballistic cycle.

Studies on secondary combustion of gun exhaust plumes.

H. Mach

Abstract

The main part of the muzzle flash of guns is due to secondary combustion of reactive propellant gases H_2 and CO with ambient air. However the mechanisms which are thought to be responsible for this reaction as well for inhibiting reactions (e.g. Carfagno's model) are rather hypothetical and not really verified by experiments on real guns. The present paper deals with the conditions leading to the ignition of secondary flash of a 7.62 mm rifle using ammunition without chemical flash suppressants. In particular we have considered the behavior of propellant gas balls included such with mixed air along their pathways from the Mach disk through the intermediate flash zone down to the location where ignition occurs. Along these lines measurements of velocities (by LDA), temperatures (by spectroscopic methods) and of oxygen contents (by use of an SF_6-tracer) were performed. Additionally temperatures were varied by using gun tubes of different lengths, oxygen content was varied by shooting into a closed vessel filled with air of variable oxygen partial pressure. Ignition was recorded by high-speed framing and streak photography. Hence ignition times could be provided.

It was found that the propellant gas/air mixing along the flow axis was rather gradually. It is considerably increased by the presence of a vortex ring. Temperature and oxygen distribution along the pathways are not uniform. Therefore it was not possible to arrive at an analytical expression for the ignition-time delay in terms of temperature T and mole fraction α of oxygen. Such an expression is well-known for the O_2/H_2 reaction. However an integral form which takes into account the time histories of T and α could be found. It includes a parameter which is equivalent to the activation energy of that reaction which controls the rate of the branching chain kinetics of the secondary combustion. The experiment yielded an activation energy which is considerably lower than the activation energy of the O_2/H_2 reaction. Thus we conclude that the governing elementary reaction for the ignition of the secondary flash is different from the elementary reaction $H + O_2 \rightarrow OH + O$ which at a higher temperature level controls the H_2/O_2 combustion. Possible other reactions will be discussed in the paper.